Proceedings of the World Congress on Engineering and Computer Science 2016 Vol II
WCECS 2016, October 19-21, 2016, San Francisco, USA

Nonlinear Finite Element Program for Reinforced Concrete Structural Control

Tawanda. Mushiri Member, IAENG, Wilson R. Nyemba, and Charles. Mbohwa

***Abstract*— Proprietary softwares are often used for structural control in engineering. Nonlinear structural control researches often uses nonlinear finite element toolbox (NLFET) code which was specifically developed for coding and simulations of structures. This paper provides an overview of the use of NLFET including the data structures and algorithms used to develop a nonlinear finite element program for reinforced concrete structural control. In order to make use of the nonlinear routines, powerful control and NLFET toolboxes, NFLET are implemented in MATLAB. The data of the structure is stored in MATLAB structures for maximum flexibility and to improve the readability of the code. Object oriented design is used to define element types so that new elements (both linear and nonlinear) can be added easily and without necessitating changes in the core analysis code.**

***Index Terms*— structural analysis, control, dynamics, NLFET toolbox, reinforced concrete**

I. INTRODUCTION

The three great virtues of a programmer are laziness, impatience, and hubris (Wall et al. 1996). Judging by the number of structural control papers asserting new or significant discoveries, it appears that the last virtue is well developed in the structural control community. However, much of the simulation in structural control has been done by writing custom software codes, indicating that the first two virtues are not developed fully among structural control programmers. Often, the unique information requirements of structural control preclude the use of proprietary structural analysis or finite element codes (Balme`s 1999). However, much could be done to save time and reuse software for numeric simulations in structural control. In addition, the recent overview of structural control by Housner et al. (1997) found that “devices and algorithms for control of non-linear systems” (emphasis in original) was a high priority of research investigation. The Nonlinear Finite Element Toolbox (NLFET) seeks to fill these needs by providing a nonlinear finite element code which uses an object-oriented approach, provides software that can be used at many levels, and licenses the software under the GNU General Public License (FSF 2002) so that the source code is freely available and redistributable.

A. DEFINITION OF THE PROBLEM

A very important issue in the design of multistoried structures is its higher performance, resistance to vibration and lighter weight structural systems. The light weight structure can be achieved with efficient design of structure but it induces large amount of vibration due to wind and earthquake forces. It is unsafe if its vibration is not controlled during earthquake and also during wind load.

B. AIM

To develop a non-linear finite element program for ease numerical simulations for vibration control of reinforced concrete structures using different control and stability methods, which ultimately reduce vibration and consequently damage to the structures.

C. OBJECTIVES

1. Develop finite element algorithm for transient dynamic analysis of reinforced concrete structures under earthquake vibration.
2. Develop feedback control algorithm for vibration control of reinforced concrete structures.
3. Carry out vibration control study under different earthquake condition.

II: LITERATURE SURVEY

A brief review of previous studies on the application of the finite element method to the analysis of reinforced structures is presented is this section. A more detailed description of the underlying theory and the application of the finite element method to the analysis of nonlinear structures is presented in excellent state- of-the-art reports by the American Society of Civil Engineers in 1982 (ASCE 1982) and 1985 (Meyer and Okamura, eds. 1985).

The earliest publication on the application of the finite element method to the analysis of RC structures was presented by Ngo and Scordelis (1967). In their study, simple beams were analyzed with a model in which concrete and reinforcing steel were represented by constant strain triangular elements, and a special bond link element was used to connect the steel to the concrete and describe the bond-slip effect.

Nilson (1972) introduced nonlinear material properties for concrete and steel and a nonlinear bond-slip relationship into the analysis and used an incremental load method of

Manuscript received March 4, 2016; revised July 23, 2016
This work was supported by the University of Johannesburg.

Tawanda Mushiri is a lecturer at the University of Zimbabwe from March 2013 to date teaching Engineering dynamics and design and is also a PhD student at the University of Johannesburg in the field of fuzzy logic systems and maintenance. Contacted at tawanda.mushiri@gmail.com / 201337963@student.uj.ac.za

Wilson Nyemba Department of Mechanical Engineering Science, Faculty of Engineering and the Built Environment, University of Johannesburg, Auckland Park 2006, Johannesburg, South Africa. Contacted at nyemba@yahoo.com

Charles Mbohwa is currently a Full Professor of Sustainability Engineering and Engineering Management at the University of Johannesburg, South Africa. Contacted at cmbohwa@uj.ac.za.

ISBN: 978-988-14048-2-4
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

nonlinear analysis. Four constant strain triangular elements were combined to form a quadrilateral element by condensing out the central node. Cracking was accounted for by stopping the solution when an element reached the tensile strength, and reloading incrementally after redefining a new cracked structure. The method was applied to concentric and eccentric reinforced concrete tensile members which were subjected to loads applied at the end of the reinforcing bars and the results were compared with experimental data.

Franklin (1970) advanced the capabilities of the analytical method by developing a nonlinear analysis which automatically accounted for cracking within finite elements and the redistribution of stresses in the structure. This made it possible to trace the response of two- dimensional systems from initial loading to failure in one continuous analysis. Incremental loading with iterations within each increment was used to account for cracking in the finite elements and for the nonlinear material behavior. Franklin used special frame-type elements, quadrilateral plane stress elements, axial bar members, two-dimensional bond links and tie links to study reinforced concrete frames and RC

For the analysis of RC beams with material and geometric nonlinearities Rajagopal (1976) developed a layered rectangular plate element with axial and bending stiffness in which concrete was treated as an orthotropic material. RC beam and slab problems have also been treated by many other investigators (Lin and Scordelis 1975; Bashur and Darwin 1978; Rots et al. 1985; Barzegar and Schnobrich 1986; Adeghe and Collins 1986; Bergmann and Pantazopoulou 1988; Cervenka et al. 1990; Kwak 1990) using similar methods.

Selna (1969) analyzed beams and frames made up of one-dimensional elements with layered cross sections which accounted for progressive cracking and changing material properties through the depth of the cross section as a function of load and time. Significant advances and extensions of the finite element analysis of reinforced concrete beams and frames to include the effects of heat transfer due to fire, as well as the time-dependent effects of creep and shrinkage, were made by Becker and Bresler (1974).

Two basically different approaches have been used so far for the analysis of RC slabs by the finite element method: the modified stiffness approach and the layer approach. The former is based on an average moment-curvature relationship which reflects the various stages of material behavior, while the latter subdivides the finite element into imaginary concrete and steel layers with idealized stress-strain relations for concrete and reinforcing steel. Dotroppe et al. (1973) used a layered finite element procedure in which slab elements were divided into layers to account for the progressive cracking through the slab thickness. Scanlon and Murray (1974) have developed a method of incorporating both cracking and time-dependent effects of creep and shrinkage in slabs. They used layered rectangular slab elements which could be cracked progressively layer by layer, and assumed that cracks propagate only parallel and perpendicular to orthogonal reinforcement. Lin and Scordelis (1975) utilized layered triangular finite elements in RC shell analysis and included the coupling between membrane and bending effects, as well as the tension stiffening effect of concrete between cracks in the model.

Even though many studies of the bond stress-slip relationship between reinforcing steel and concrete have been conducted, considerable uncertainty about this complex phenomenon still exists, because of the many parameters which are involved. As a result, most finite element studies of RC structures do not account for bond-slip of reinforcing steel and many researchers express the opinion that this effect is included in the tension-stiffening model.

Very little work has been done, so far, on the three-dimensional behavior of reinforced concrete systems using solid finite elements, because of the computational effort involved and the lack of knowledge of the material behavior of concrete under three- dimensional stress states. Suidan and Schnobrich (1973) were the first to study the behavior of beams with 20-node three-dimensional isoparametric finite elements. The behavior of concrete in compression was assumed elasto-plastic based on the Von-Mises yield criterion. A coarse finite element mesh was used in these analyses for cost reasons.

In spite of the large number of previous studies on the nonlinear finite element analysis of structures, only few conclusions of general applicability have been arrived at. The inclusion of the effects of tension stiffening and bond-slip is a case in point. Since few rational models of this difficult problem have been proposed so far, it is rather impossible to assess exactly what aspects of the behavior are included in each study and what the relative contribution of each is. Similar conclusions can be reached with regard to other aspects of the finite element analysis. Even though the varying level of sophistication of proposed models is often motivated by computational cost considerations, the multitude of proposed approaches can lead to the conclusion that the skill and experience of the analyst is the most important aspect of the study and that the selection of the appropriate model depends on the problem to be solved.

Recognizing that many of the previously proposed models and methods have not been fully verified so far, it is the intent of this study to address some of the model selection issues, in particular

III: METHODOLOGY

FEM has become a standard procedure for the analysis of structures. In general, to obtain reasonable accuracy, a structure has to be modeled with much finer mesh leading to a large number of degrees of freedom. This increases the problem size as well as the computational time and cost. Since the control system of structure involves multiple solutions in iteration it becomes impossible to solve complex problems within reasonable time and at an affordable cost with available serial computing resources. High performance computers provide the necessary supercomputing power and enable the designer to test larger problems by expediting the analysis process.

A. OBJECT-ORIENTED DESIGN APPROACH

The Element definition in NLFET is object-oriented that is the code for each element provides the element stiffness matrix and all other information needed by the analysis engine. To add elements to NLFET designer simply writes a new subroutine which conforms to the application programming interface (API). The element subroutines can be separated from the analysis code since it is easy to use different analysis engines for specific problems, and changes to the analysis engine are not required when adding new elements. Though the MATLAB scripting language is not specifically designed for object-oriented programming it allows quick development and is particularly adapted to numerical computation.

To demonstrate the use of NLFET a single bay frame mass. The elements are within squares and the node are circled. The column elements 7 and 8 are assumed to remain elastic so they are modelled using Bernouli beam elements. Element 9 uses a nonlinear beam element which can undergo bilinear hysteric yielding at the ends, as idealized by plastic hinges near 8 and 9. Element 10 is a viscous damper to provide passive control of the frame.

Data pertaining to the structure is stored in a MATLAB structure variable named data in our particular case. The structure, a single-bay frame with associated mass will be given

B. DEFINING MATERIAL AND SECTION PROPERTIES

NLFET separates the definition of material and section properties for maximum flexibility and coding efficiency. Material definitions will be stored as an array of structures and added to the data structure with a code (all units are inches and pounds).

```
% Steel 37 ksi
data.materials(7).E = 30000000; data.materials(7).Fy = 37000;
data.materials(7).name = '37ksi steel';
data.materials(7).alpha = 0;
```

Where,

E is Young's modulus,

Fy is the yield stress,

name is the name of the material, and alpha is the coefficient of the post- yielding incremental stiffness of the material (setting it to zero here indicates the material is perfectly plastic). The damper element material requires only the damping coefficient, *c*, in the material definition:

```
data.materials (8).c = 479.0;
```

The value for c used here is chosen so that the effective damping of the structure increases to 5% ($\varepsilon = 0.05$) from an uncontrolled value of 2%. The columns use a W15x110 section which is defined in the data structure with the following code:

```
% W15x110 (columns)
data.sections(7).A=33.0;
data.sections(7).I=1241;
data.sections(7).name = 'W15x110';
```

where A is the cross-sectional area and I is the moment of inertia. In this case the beam uses a W22x67 section:

```
% W22x067 (beam)
data.sections(8).A=20.2;
data.sections(8).I=1840;
data.sections(8).Z=178.0;
data.sections(8).name = 'W22x67';
```

Note that because the beam can yield, a plastic modulus (Z) must be given.

C. ELEMENT DEFINITION

With the materials and sections defined, the definition of the elements is done with the following code:

```
data.elements(7).material = 7;
data.elements(7).section = 7;
data.elements(7).nodes = [ 7 8];
data.elements(7).type = 'beam8d';
data.elements(8) =
 data.elements(7);
data.elements(8).nodes = [ 10 3];
data.elements(9).material = 7;
data.elements(9).section = 8;
data.elements(9).nodes = [ 8 9];
data.elements(9).type = 'nlbeam8d';
data.elements(10).material = 8;
data.elements(10).nodes = [ 7 9];
data.elements(10).type = 'vdamper8d';
```

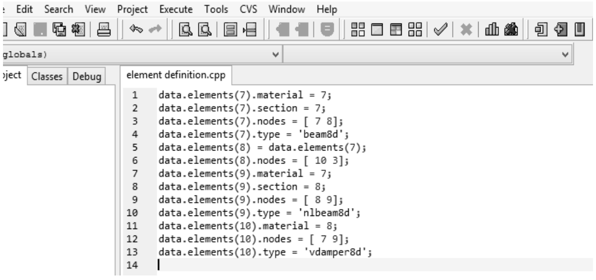
The material and section are the indexes into the materials and sections array, respectively. The nodes field is a connectivity vector for the element, and type indicates the element type (a corresponding .m file must be defined: beam2d.m, nlbeam2d.m, etc.)

IV: SIMULATION RESULTS

The simulation took 15 seconds and it includes a linear analysis for comparison with the results of the nonlinear analysis. The MATLAB lsim command was used for the linear analysis, and the ODE suite of nonlinear, differential equation solvers (Shampine and Reichelt, 1997) is used for the nonlinear analysis. Both analysis algorithms require a state space formulation so that second-order differential equations of motion must be converted to first-order differential equations. This results in a state space system with six states $X, \theta_2, \theta_3, \gamma, \vartheta_1, \vartheta_2$. In addition, the nonlinear analysis requires two additional states, α_1 and α_2 which represent the angle of rotation "inside" the plastic hinges at nodes 8 and 9, respectively. Fig 4 shows the rotation at node 2 (θ_1) for the linear and nonlinear analysis. The rotation on the inside of the plastic hinge (θ_2) is also plotted in fig. 4. The "plateaus" in the θ_2 state indicates that the beam is yielding at the plastic hinges. The total run time for the nonlinear case was over 11 minutes compared to 0.23 seconds for the linear analysis [12, 13]. Thus the nonlinear analysis is over 2800 times slower, indicating that a different solution algorithm (i.e. Newmark-Beta) may be more efficient.

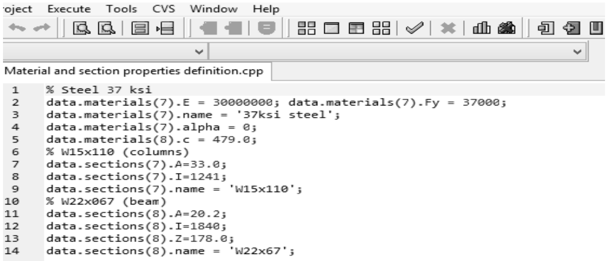
V: CONCLUSION

The nonlinear finite element toolbox has been developed to ease numerical simulation in structural control. Advantages of this software are the object-oriented design, overall modular architecture, and the availability of the source code. A simple dynamic analysis example has been presented to illustrate the use of NLFET.



```
data.elements(7).material = 7;
data.elements(7).section = 7;
data.elements(7).nodes = [ 7 8];
data.elements(7).type = 'beam8d';
data.elements(8) = data.elements(7);
data.elements(8).nodes = [ 10 3];
data.elements(9).material = 7;
data.elements(9).section = 8;
data.elements(9).nodes = [ 8 9];
data.elements(9).type = 'nlbeam8d';
data.elements(10).material = 8;
data.elements(10).nodes = [ 7 9];
data.elements(10).type = 'vdamper8d';
```

Fig 1: ELEMENT DEFINITION CODE IN NLFET



```
% Steel 37 ksi
data.materials(7).E = 30000000; data.materials(7).Fy = 37000;
data.materials(7).name = '37ksi steel';
data.materials(7).alpha = 0;
data.materials(8).c = 479.0;
% W15x110 (columns)
data.sections(7).A=33.0;
data.sections(7).I=1241;
data.sections(7).name = 'W15x110';
% W22x067 (beam)
data.sections(8).A=20.2;
data.sections(8).I=1840;
data.sections(8).Z=178.0;
data.sections(8).name = 'W22x67';
```

Fig 2: MATERIAL AND SECTION PROPERTIES DEFINITION CODE IN NLFET

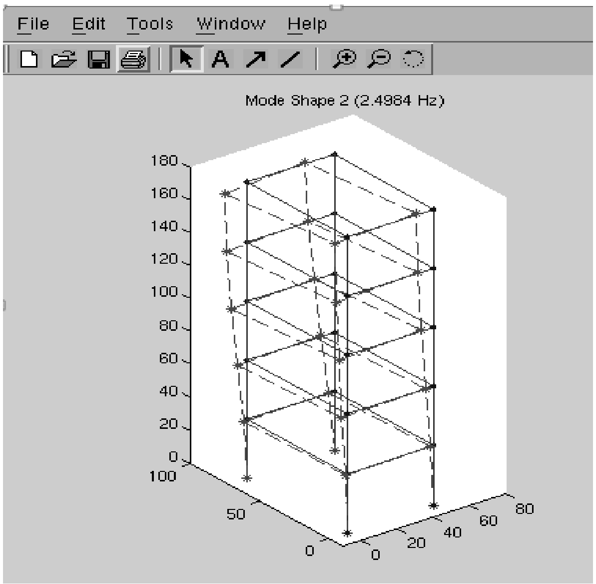


Fig 3: MODE SHAPE DIAGRAM

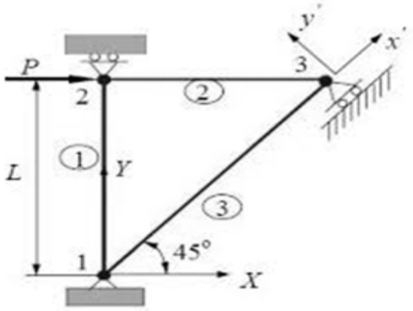


Fig 4: DIAGRAM OF NODES

ACKNOWLEDGMENTS

I would like to thank my supervisor Professor Mbohwa for support and supervision. University of Johannesburg paid me all my trip and accommodation expenses.

REFERENCES

[1] David W Knowles IV; Nader Jalili 'Structural vibration control using an active resonator absorber: modeling and control implementation' Journal of Smart Materials and Structures, October 2004, vol. 13, no. 5, pp. 998-1005(8)

[2] Do-Hyung Kim, Jae-Hung Han, Dae-Hyun Kim and In Lee 'Vibration control of structures with interferometric sensor non-linearity' journal of Smart Material and Structure 13 92-99, 2004

[3] Young-Hun Lim 'Finite-element simulation of closed loop vibration control of a smart plate under transient loading' Journal of Smart Material and Structure 12 272-286 2003.

[4] Wei Liu and Zhikun Hou , 'model reduction in structural vibration control and its application' ,proceeding 16th ASCE Engineering Mechanics Conference ,USA, Seattle July 16th –18th , 2003,

[5] A. P. Kumbhar, M. S. Shah and B. Dattaguru, "Three Dimensional Fracture Analysis Using Parallel Algorithms", International conference on Advanced structural Integrity , Bangalore , India. July 14-17 2004

[6] Shah M.S., Shembekar P.S. and Naik N.K., *'Prediction of notched strength of composites using finite element analysis and fracture mechanics'* paper presented at the 5th National Convention of Aerospace Engineers held at Chandigarh, India, on 22-23 Feb. 1990.

[7] M.S. Shah and T. Kant, "Finite Element Analysis of Fibre-Reinforced Polymer Shells Using Higher-Order Shear Deformation Theories on Parallel Distributed Memory Machines", Int. J. of Comp. Applications in Technology. Vol 12, Nos. 2/3/4/5, 1999, p206

[8] B. Datta and D. Sarkissian, "A Computational Method for Feedback Control in Distributed Parameter Systems",Proceedings of the 8th IEEE Int. Conference on Methods and Models in Robotics and Automation, pp 139-144, 2002

[9] B. Datta and D. Sarkissian,"Computational Methods for Feedback Control in Damped Gyroscopic Second-order Systems", Proc. IEEE International Conference on Decision and Control , Las Vegas, NV, Dec 2002. [10] B. Datta, "Finite Element Model Updating, Eigenstructure Assignment and Eigenvalue Embedding Techniques for Vibrating Systems", Mechanical Systems and Signal Processing , (Invited paper), special issue on Vibration Control, vol 16, nr 1, 83-96, 2001.

[11] Brij N Agrawal and Hong-Jen Chen," Algorithms for active vibration isolation on spacecraft using a Stewart platform",Journal of Smart Material and Structures, 13 No 4 (August 2004) 873-880

[12] Potts, J.F. and Oler, J.W., Finite Element Applications with Microcomputers, Prentice Hall, 1989.

[13] Rao, S.S., Finite Element Method in Engineering, 3rd Ed., Butterworth-Heinemann, 1998.

[14] Reddy, J.N., Energy and Variational Methods in Applied Mechanics, John Wiley, 1984.

[15] Reddy, J.N., An Introduction to the Finite Element Method, 3rd Edition, McGraw-Hill, 2006.

[16] Ross, C.T.F., Finite Element Methods in Engineering Science, Prentice-Hall, 1993.

[17] Shabana, A.A., Comuputational Continuum Mechanics, Cambridge Univ. Press, 2008.

[18] Stasa, F.L., Applied Finite Element Analysis for Engineers, Holt, Rinehart and Winston, 1985.

[19] Zienkiewicz, O.C., The Finite Element Method: From Intuition to Generality, Applied Mechanics Reviews, Vol. 23, pp. 249-256, 1970.

[20] Zienkiewicz, O.C., Taylor, R.L.and Zhu, J.Z., The Finite Element Method: It's Basis and Fundamentals, Volume 1, 6th Edition, McGraw-Hill, 2005.